International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 9, Issue 4, Aug 2019, 943-954

THE CFD ANALYSIS OF SUBSONIC FLOW AROUND STRUTS OF AIRFOIL AND CYLINDRICAL SHAPE ATTACHED TO A CONICAL DIFFUSER AT EXHAUST OF A GAS TURBINE ENGINE

VINOD. L[1], MAHENDRA. M. A[2] & MAHANTAYYA. K. H[3]

[1,2]*Assistant Professor, Department of Aeronautical Engineering, NMIT, Bengalur, India*

[3]*Assistant Professor, Department of Aeronautical Engineering, AIT, Bengaluru, India*

ABSTRACT

After burners are employed in an aero engine of a fighter aircrafts, to achieve maximum thrust during take-off, climb and during some special combat maneuvers. Since the fuel consumption is high during the afterburning operation it is employed only when necessary. The diffuser duct is the first component of the afterburner system, situated immediately next to the exit of the low-pressure turbine. The struts are the supporting structures, which connect the parts of diffuser duct. These struts will guide the flow in the diffuser duct. The presence of the struts will cause some pressure losses. The flow is decelerated in the diffuser duct. The Mach number also reduces along the length of the diffuser. The design of the diffuser duct should be made so that the recirculation should be minimum at the exit of the diffuser and the thermodynamics properties at the exit should meet the requirement for the proper mixing of the fuel and air with the hot gases passing through the diffuser & hence ensures complete combustion in the upstream zones of the diffuser. The flow inside an afterburner is highly complex due to the presence of the diffuser, flame stabilizer, variable area nozzle and the interaction between core and bypass flow through anti-screech holes. Therefore the performance of afterburner depends to a great extent on the internal aerodynamics. Experimental analysis of such flow is a difficult task.

Cost and time involved for optimization design are very high. Computational fluid dynamics (CFD) provides a cost effective solution in analyzing. Its suitability to support the combustion process for the design of short efficient reheats system. Hence in this study, the flow in the diffuser duct in details by carrying out the analysis of subsonic flows in a three-dimensional model of an afterburner duct using CFD. The flow in the bypass region of the duct is considered, as it does not have any impact on diffuser performance.

In the current study two stages viz. diffuser duct with struts and without struts were carried out and for both cases the pressure recovery, total pressure loss, essential flow properties like Mach number, velocity, statics pressure, swirl are compared for both the cases. Considering the periodicity of geometry, analysis has been carried out using 45-degree sector model of the diffuser duct with and without struts. Using ICEM-CFD pre-processor an unstructured grid has been generated and Navier-stroke equation is applied to carry out using ANSYS CFX CFD software, flow simulated by solving governing equation of mass, momentum, energy. Turbulence closure is achieved with k-epsilon turbulence model with standard wall functions.

KEYWORDS: Low-Pressure Turbine, CFD Analysis & Airfoil

Received: Jun 01, 2019; **Accepted:** Jun 20, 2019; **Published:** Jul 13, 2019; **Paper Id.:** IJMPERDAUG201996

Original Article

INTRODUCTION

Simple jet engine with a diffuser is shown in figure 1, which works on Brayton cycle. The air is sucked inside the engine through the fan and it is compressed to the required pressure in the compressor. The compressed air is passed to the combustion chamber where the fuel and air gets mixed and burnt. Then the combusted products are expanded through the turbine. Hot gases continue in turbine to expand & blast out through high velocity and forces.

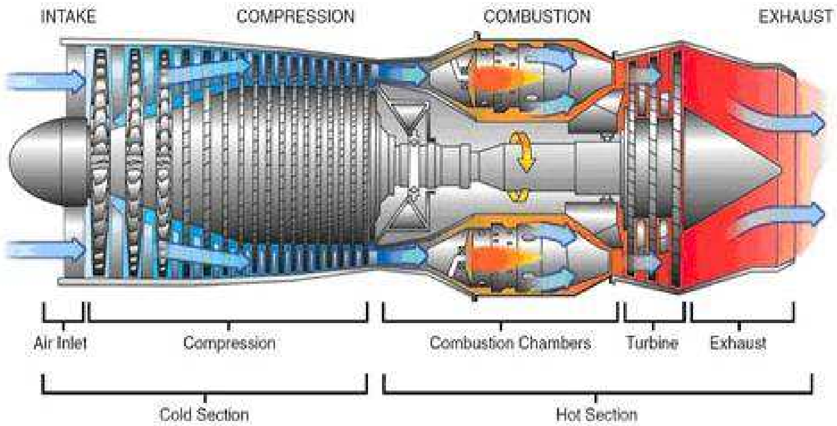

Figure 1: Shows Detailed Jet Engine with Diffuser

The gas turbine engine has high efficiency and good performance. Most of the aircraft are incorporated with turbojet, turbofan, or turboprop engines. The afterburners are mostly incorporated in turbojet and turbofan gas turbine engines. The afterburner is kept downstream of the turbine. It uses the unburned oxygen in the exhaust gases coming from the turbine. Extra fuel is added resulting in increase in fluid temperature. When this burned gas is expanded through the nozzle, it results in high velocity exhaust jet, which boosts the engine thrust.

OBJECTIVES

The design requirements of an afterburner depend to a greater extent on the internal aerodynamics of flow. Hence, research efforts are directed to improve the aerodynamics of flow in an afterburner. Computational methods have emerged in recent years for accurate predictions of complex flows. The residence time available for the fuel preparation, mixing and burning is reduced because of high speed flow in after burner. In order to increase the residence time of the hot gases, which are coming out from the low pressure turbine, the flow is decorated in diffuser duct. This is placed immediately next to the low-pressure turbine exit. Air foiled shaped struts support the inner and outer wall of the diffuser besides guiding the flow. The presence of struts causes some pressure losses.

Present study deals with the implementation of CFD for three-dimensional analysis of flow in an afterburner diffuser duct. CFX-CFD software packing has been used for the analysis.

The present study has been carried out in following steps.

1. Subsonic flow analysis in an afterburner diffuser duct without struts.

2. Subsonic flow analysis in an afterburner diffuser duct with NACA struts (symmetric), cylindrical strut and elliptical strut.

3. The flow in the bypass region of the afterburner is neglected in this study as it does not have any impact on the performance of the afterburner.

4. The recirculation zone at the exit of the diffuser duct, velocity distribution, Mach number, total pressure loss, effect of swirl, impact of pressure recovery on the flow are the main parameters of interest in this study.

5. The presence of struts will cause total pressure loses in diffuser. To analyses the effect of the struts on the variables. Simulation are made first with an afterburner duct without and with struts.

BRIEF REVIEW OF LITERATURE

Useller, James W [1] presented the results of several full-scale turbojet-engine afterburner investigations that had been conducted by NASA. Considering combustion efficiency as a criterion of the afterburner performance effects of combustion chamber lengthen afterburner combustion performance was experimentally determined. The test facility included a conical annular diffuser a two ring V-gutter flame holder fixed area exhaust nozzle and external manifold fuel system representing a practical afterburner unit.

Ganesan [2] carried out theoretical studies on turbulent flow in a conical baffled with circular duct. The flow is two dimensional, axisymmetric, steady, turbulent and chemically non-reacting. For a particular inlet velocity, axial and radial velocities along with kinetic energy and length scale of turbulence behind the bluff body have been predicted at various axial locations.

METHODOLOGY ICEM-CFD

- Importing the Geometry
- Repairing the Geometry
- Parts Creation
- Assigning Mesh Sizes
- Generating the Hexahedral Mesh with Smooth Transition
- Exporting the Mesh Diffuser with Airfoil Strut, cylindrical Strut and Elliptical Strut Modelling

ANSYS CFX-CFD

- CFX-PRE (physical pre-processor)
- CFX-solver (solver)
- CFX-post (post manager)

COMPUTATIONAL DETAILS OF PRESENT WORK

This chapter contains the geometric and design details of the afterburner diffuser duct with NACA 0012 aerofoil strut, cylindrical strut details regarding analysis of problem under consideration i.e. grid generation, the prescribed fluid properties and boundary conditions employed for the computations.

Geometric Modelling

The diffuser duct is designed by using the CATIA V5 R16 software. The diffuser with 8 NACA 0012 airfoil and cylindrical struts is designed at 45-degree sector on diffuser duct. So that we can take one strut for CFD analysis. The duct is extended for one meter to avoid flow drop at the outlet of diffuser. The dimensions of diffuser duct are taken from [20]. Figure 2 shows Geometry of the Diffuser duct. [20]

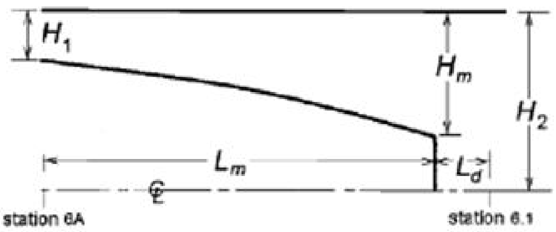

Figure 2: Shows Geometry of Diffuser

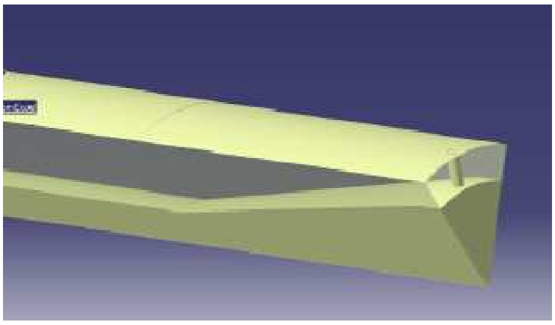

Figure 3: Shows CATIA Model of Diffuser with Strut

The after burner geometry is generated in CATIA V5 R16 CAD software and imported as an IGES file to ICEM – CFD. After importing to ICEM-CFD following steps are done

- First the closed surface sets are created followed by creation of domain. The whole model is made as a single domain.
- The body fitted O-grid is generated and domain is discretized with hexahedral elements.
- Once grid is generated is over the model is saved as DTF (data transfer file) from intended to be used in CFX-CFD to specify the boundary condition and to solver.

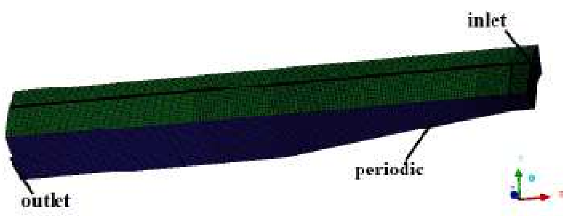

Figure 4: Shows Meshed Model of Diffuser with Strut

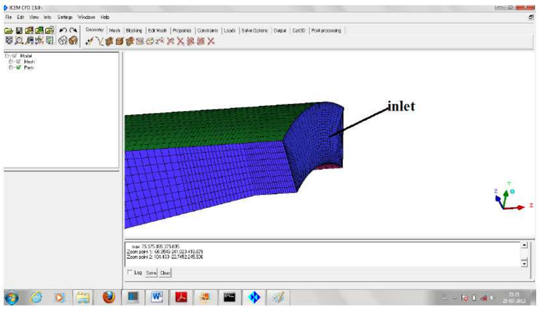

Figure 5: Shows Meshed Model of Diffuser without Strut

RESULTS AND DISCUSSION

Subsonic flow inside the afterburner diffuser duct has been successfully simulated using the ANSYS CFX-CFD software. The present chapter gives a detailed account of the results obtained including the plots of thermodynamics properties viz, static pressure, total pressure, velocity, Mach number, total temperature etc.

Flow Analysis for Afterburner Diffuser Duct without &with Struts.

That the total pressure decreases along the diffuser duct, which is due to the skin friction effect.

Figure 12 to 24 shows the plots of total pressure, velocity vectors etc. It can be seen that the velocity further decrees from the inlet to the out along the diffuser dude to the presence of struts. Total pressure losses will be more which is because of the increased skin friction coefficient along the surface of struts. It is observed that the presence of struts increases the turbulent K.E of the flow in the diffuser duct.

Table 1: After Burner Diffuser Duct Without Struts

Sl. No.	Parameters	Unit	Experimental	CFD
1	Mass flow rate	Kg/sec	12.9642	12.9642 2
2	Absolute Total pressure	Pascal	525249	525003
3	Absolute static pressure	Pascal	513920	512809
4	Total temperature	K	686	686.50
5	Mach Number	Mach	0.1817	0.1642

Flow Analysis for Afterburner Diffuser Duct Without Struts

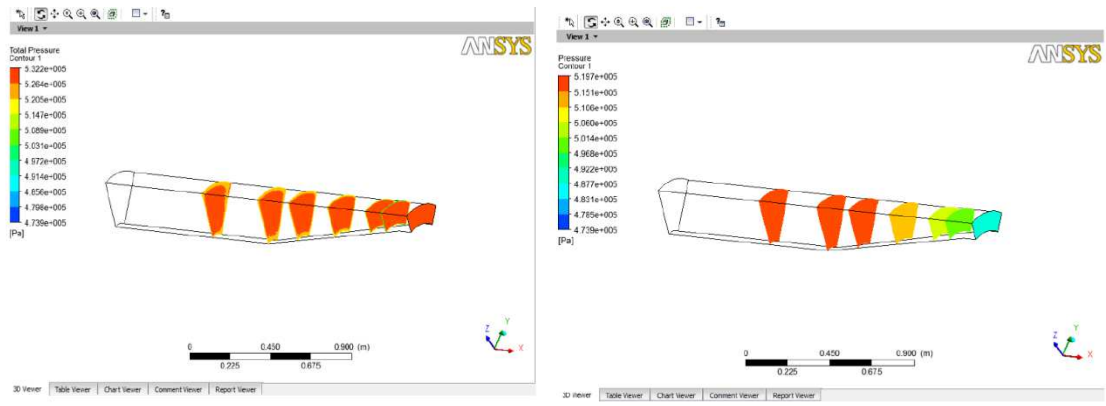

Figure 6: Contours of Total Pressure &Static Pressure

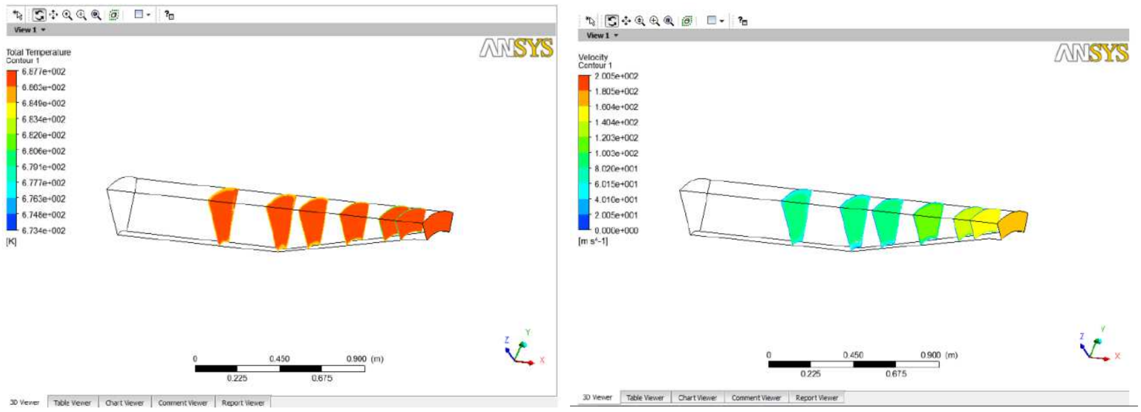

Figure 7: Contours of Total Temperature & Velocity

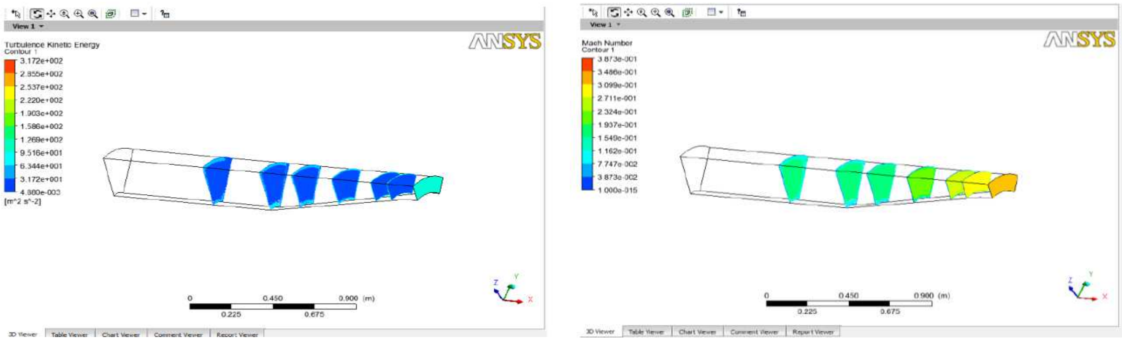

Figure 8: Contour of Turbulence Kinetic Energy & Mach Number

Flow Analysis for Afterburner Diffuser Duct with Airfoil Struts of Zero, Four & Eight Degree Swirl

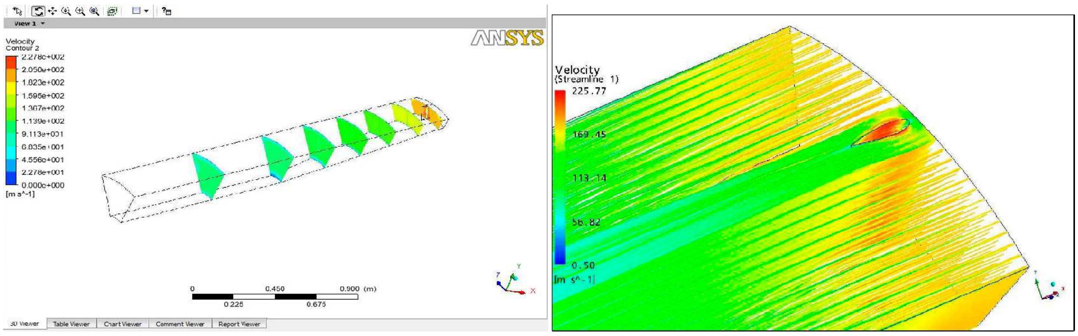

Figure 9: Shows Velocity & Velocity Near Airfoil Strut Contours

FOR 4 DEGREE SWIRL

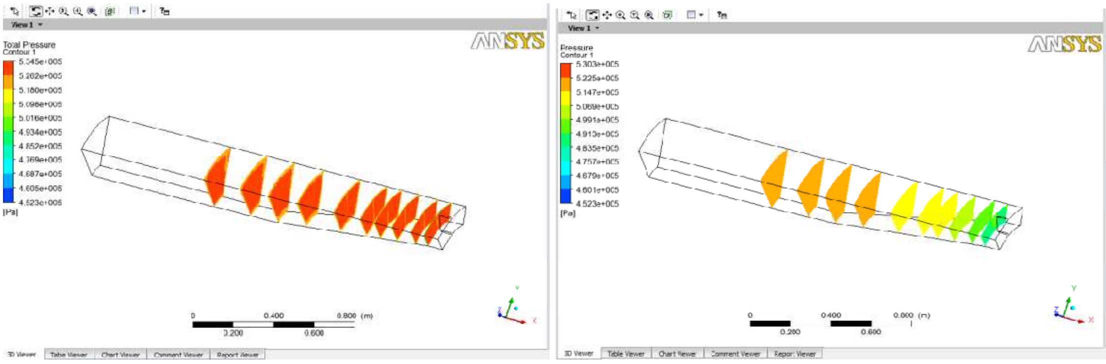

Figure 10: Shows Total Pressure & Pressure Contours

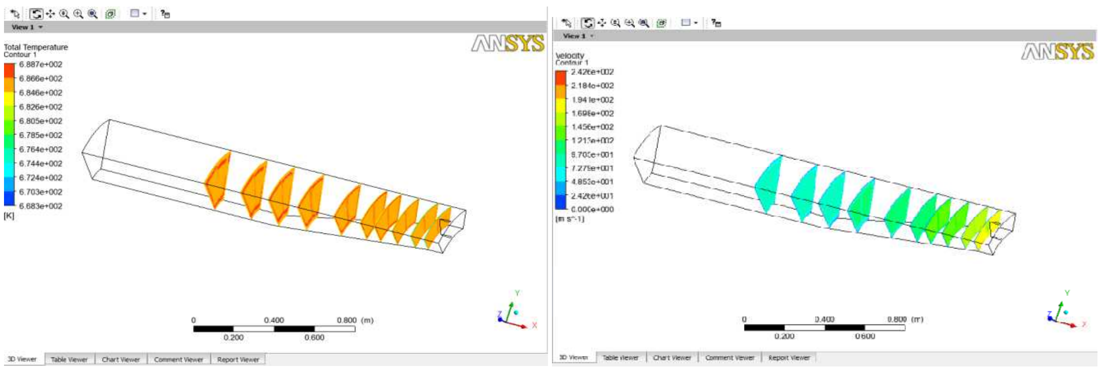

Figure 11: Shows Temperature & Velocity Contours

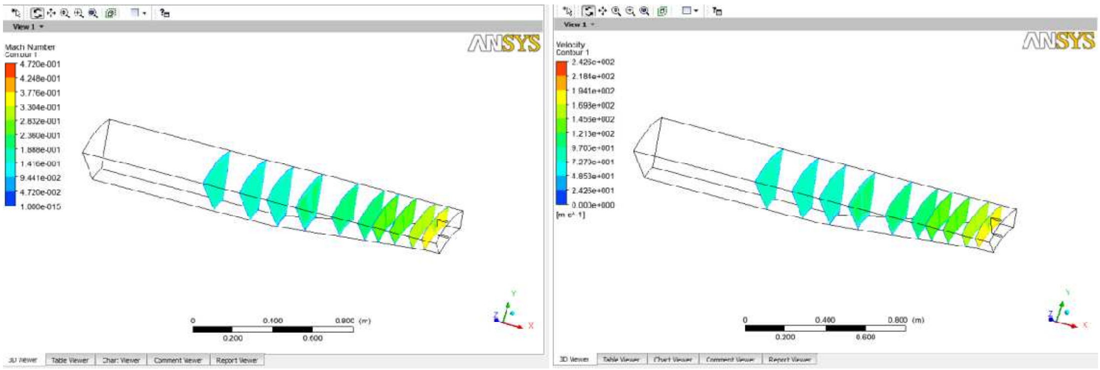

Figure 12: Shows Mach Number, Kinetic Energy Contours

FOR 8 DEGREE OF SWIRL

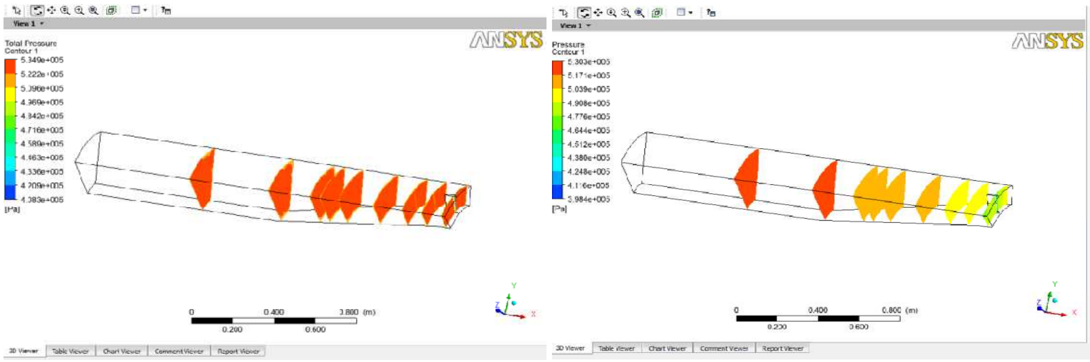

Figure 13: Shows Total Pressure & Pressure Contours

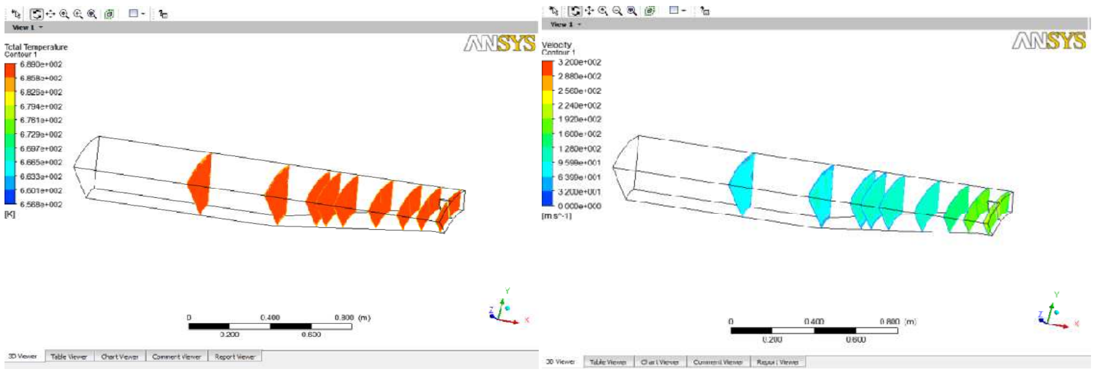

Figure 14: Shows Temperature & Velocity Contours

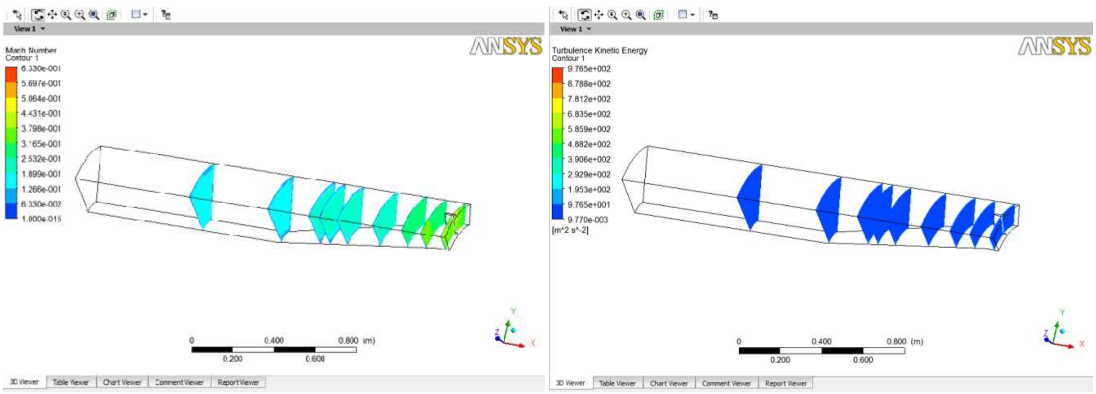

Figure 15: Shows Mach Number, Kinetic Energy Contours

DUCT WITH CYLINDRICAL STRUTS FOR 0^{O}, 4^{O}, 8^{O} SWIRL

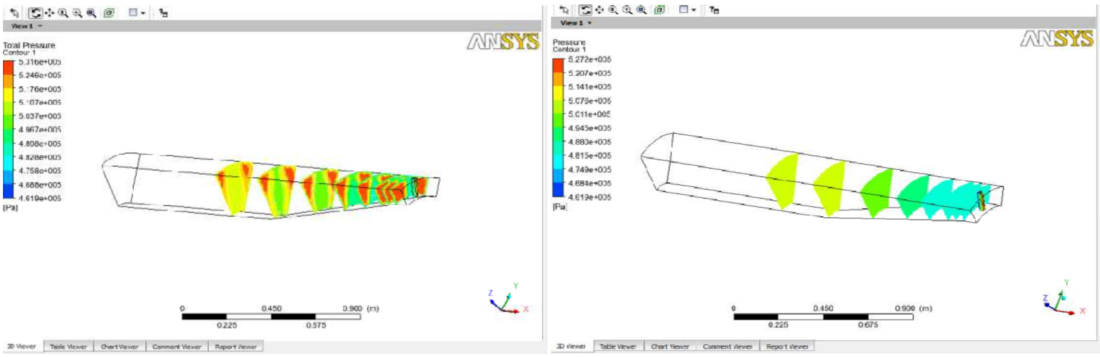

Figure 16: Shows Total Pressure & Pressure Contours

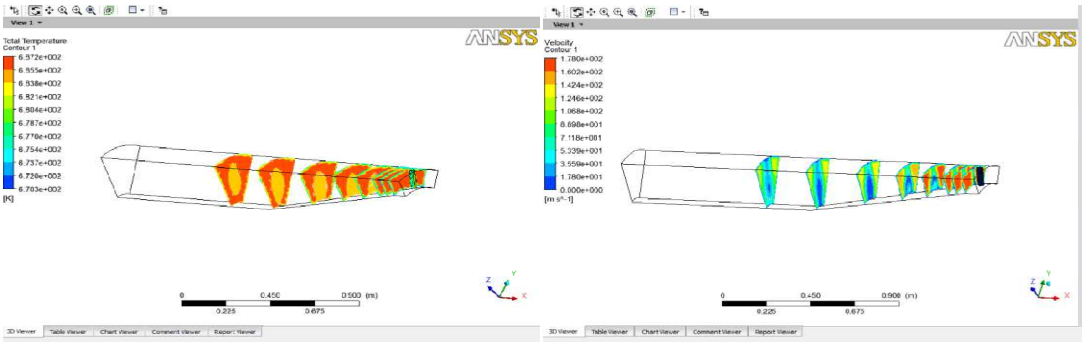

Figure 17: Shows Temperature &Velocity Contour

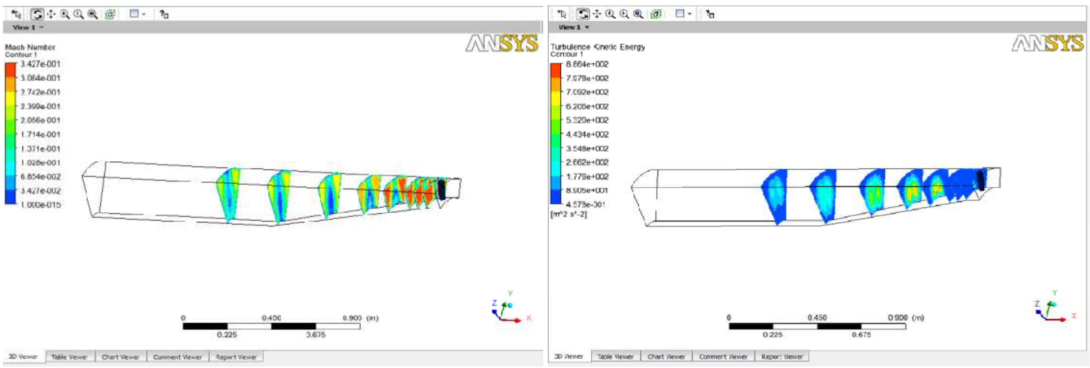

Figure 18: Shows Mach Number, Kinetic Energy Contours

For 4 Degree of Swirl

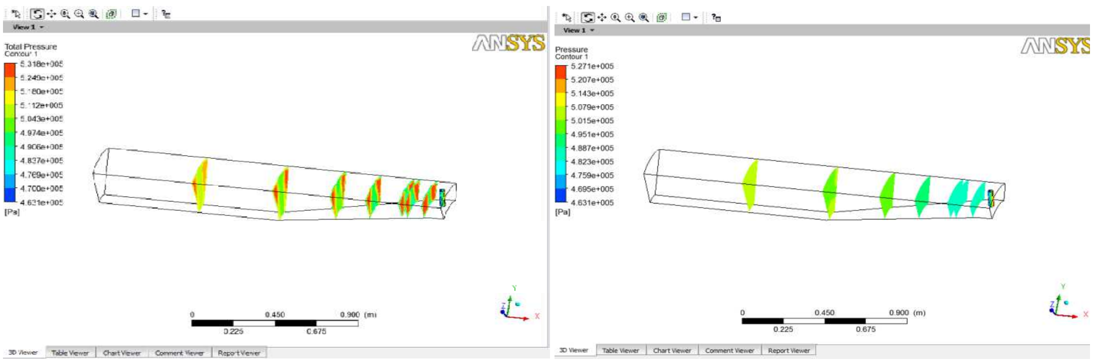

Figure 19: Shows Total Pressure, Pressure Contours

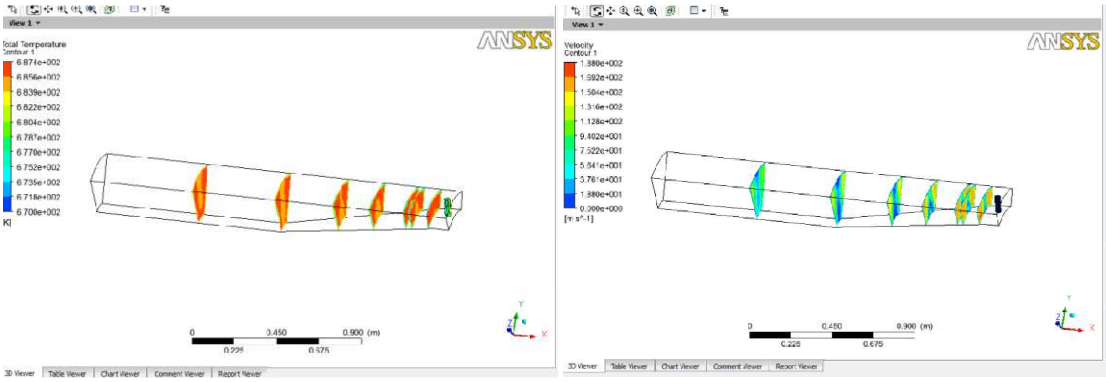

Figure 20: Shows Temperature, Velocity Contours

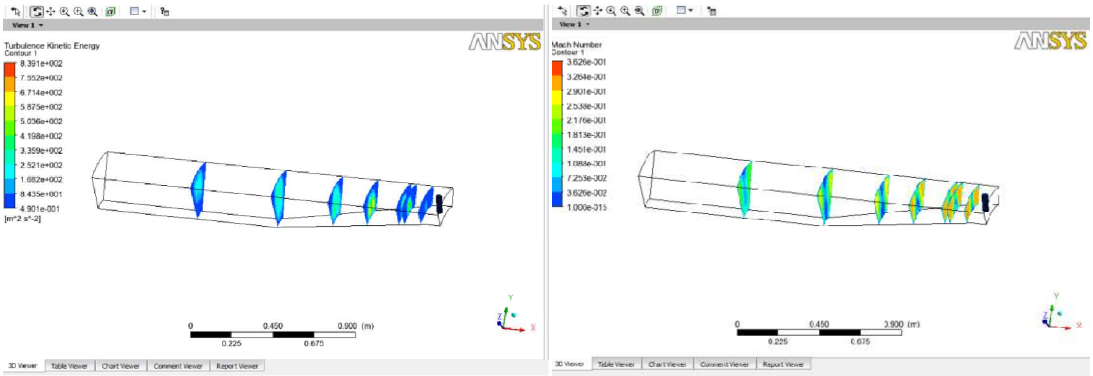

Figure 21: Shows Mach Number, Kinetic Energy Contour

For 8 Degree of Swirl

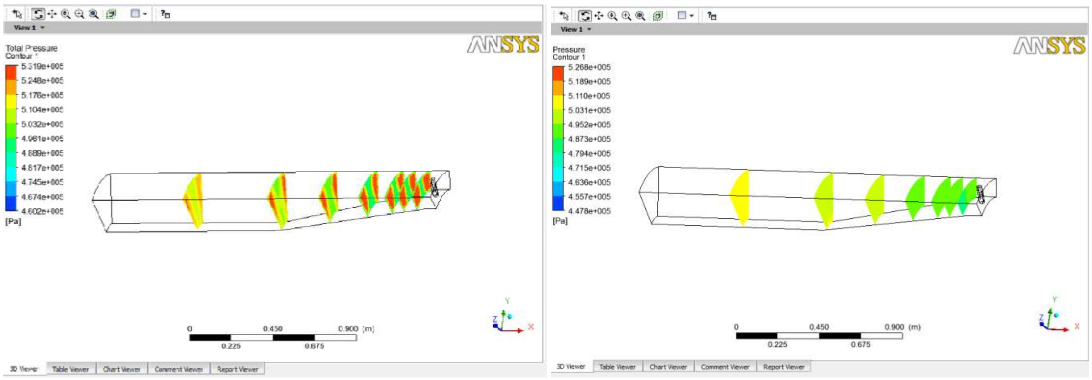

Figure 22: Shows Total Pressure, Pressure Contours

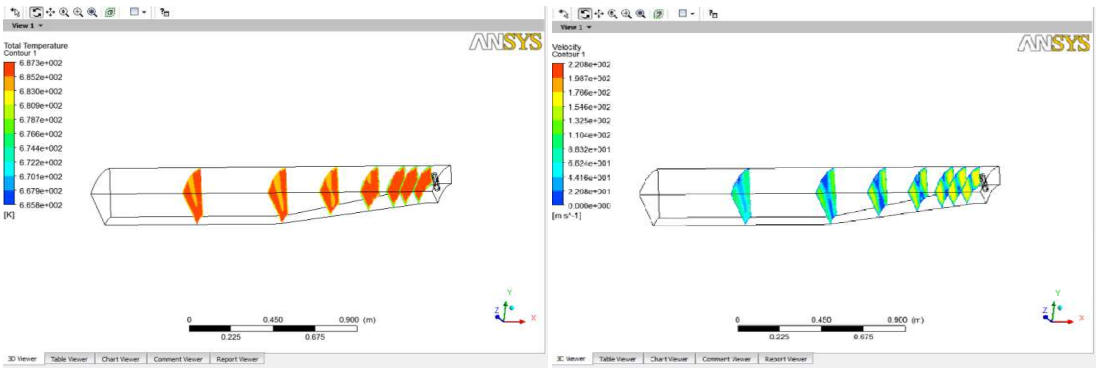

Figure 23: Shows Temperature, Velocity Contours

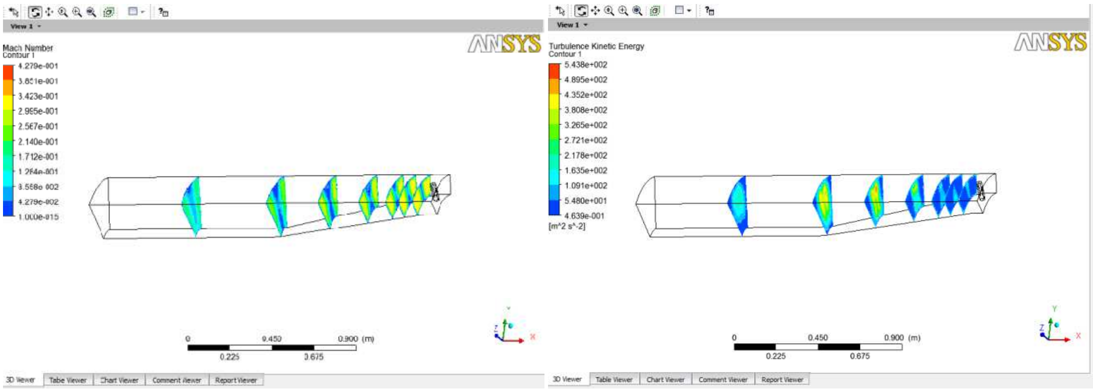

Figure 24: Shows Mach Number & Kinetic Energy Contours

Table 2: Shows Results for NACA 0012 Airfoil Struts

Sl. No.	Cases	Variables	Value		% of Total Pressure Loss	% of Total Pressure Recovery
			Inlet	**Outlet**		
1	Diffuser without strut	Total pressure (Pa)	529924	525003	0.92	-0.711
		Absolute pressure(Pa)	470553	512809		
		Velocity(m/sec)	237.17	81.35		
		Mach number	0.4249	0.1642		
		Total temperature(K)	686.5	686.50		

The total pressure loss with struts is 7292 Pascals. This higher value of pressure drop is due to the skin friction of the strut surfaces. Airfoil struts contribute to an additional pressure loss of 2617 Pascals. A pressure loss of 2617 Pascals is practically low because the flow around the strut is attached to the strut surface and there is no flow separation from the strut wall.

Table 3: Computed Results for the Simulation of Diffuser with Airfoil Struts

Sl. No	Cases	Variables	Value		% of Total Pressure Loss	% of Total Pressure Recovery
1	Diffuser with Airfoil strut (zero degree swirl)		**Inlet**	**Outlet**		
		Total pressure (Pa)	529924	524632	0.99	-0.55
		Absolute pressure(Pa)	470553	504134		
		Velocity(m/sec)	237.17	73.68		
		Mach number	0.4249	0.171		
		Total temperature(K)	686.5	686.95		
2	Diffuser with Airfoil strut (4 degree swirl)		**Inlet**	**Outlet**		
		Total pressure (Pa)	529924	523767	1.16	-0.57
		Absolute pressure(Pa)	470553	504753		
		Velocity(m/sec)	237.17	76.73		
		Mach number	0.4249	0.165		
		Total temperature(K)	686.5	686.67		
3	Diffuser with Airfoil strut (8 degree swirl)		**Inlet**	**Outlet**		
		Total pressure (Pa)	529924	522716	1.36	-0.59
		Absolute pressure(Pa)	470553	518729		
		Velocity(m/sec)	237.17	78.24		
		Mach number	0.4249	0.178		
		Total temperature(K)	686.5	686.89		

Table 4: Shows the Computed Results for the Simulation of Diffuser with Cylindrical Strut, For 0, 4 and 8 Degrees of Swirl.

Sl. No.	Cases	Variables	Value		% of Total Pressure loss	% of Total Pressure recovery
1	Diffuser with cylindrical strut (zero degree swirl)		**Inlet**	**Outlet**		
		Total pressure (Pa)	529924	525713	0.79	-0.77
		Absolute pressure(Pa)	470553	517858		
		Velocity(m/sec)	237.17	80.35		
		Mach number	0.4249	0.139		
		Total temperature(K)	686.5	711		
2	Diffuser with cylindrical strut (4 degree swirl)		**Inlet**	**Outlet**		
		Total pressure (Pa)	529924	523796	1.156	-0.79
		Absolute pressure(Pa)	470553	517642		
		Velocity(m/sec)	237.17	78.42		
		Mach number	0.4249	0.119		
		Total temperature(K)	686.5	689.8		
3	Diffuser with cylindrical strut (8 degree swirl)		**Inlet**	**Outlet**		
		Total pressure (Pa)	529924	523369	1.425	-0.79
		Absolute pressure(Pa)	470553	517471		
		Velocity(m/sec)	237.17	82.33		
		Mach number	0.4249	0.105		
		Total temperature(K)	686.5	698.87		

The total pressure loss, pressure recovery are calculated. The total pressure loss with struts is 5292 Pascals. This higher value of pressure drop is due to the skin friction of the strut surfaces. Cylindrical struts contribute to an additional pressure loss of 371 Pascals. A pressure loss of 371 Pascals is practically low because the flow around the strut is attached to the strut surface and there is no flow separation from the strut wall.

CONCLUSIONS

The present investigation is aimed in detail the effect of swirl angle and struts on the flow properties and its suitability to support the combustion process for the design of short efficient reheat system, as the flow inside an afterburner is highly complex and three dimensional with a high speed flow, reducing time for mixing of fuel and air. The following conclusion are drawn from this,

- Results have been tabulated for diffuser without and with airfoil 0012 & cylindrical strut.
- The contribution of NACA 0012 airfoil struts& cylindrical struts for pressure loss and pressure recovery is estimated.
- The airfoil struts are the better when compared with cylindrical.
- As Increase in the turbulence of flow, its lead to optimum mixing of fuel and air in diffuser duct.

REFERENCES

1. *Useller, James W. (1959), Effect of combustor length on afterburner combustion. Lewis Research centre-NASA*
2. *V. Ganesan, Numerical computation of turbulent flow behind a conical baffle Indian journal of technology, vol-18, pp.447-450, Nov-1980.*

3. *S.V.Unaune and V. Ganesan, Analysis of reacting flows in an aero engine afterburning using computational fluids, Indian journal of Engineering and materials sciences, vol-11, pp31-37, Feb-2004.*

4. *Dr.Isaac,J.J, Rajashekar.C, N.R., Ramesh, et al, Afterburner flow visualization studies in a water tunnel, CABE-paper-1992*

5. *M.Rajkumar and V. Ganesan, Analysis of Non-Reacting flow in an aircraft gas turbine engine afterburner model using finite volume method, Indian journals of engineering sciences, Vol-10, pp.341-352, October 2003.*

6. *Stefano Ubertini, Umberto Desideri, E Experimental performance analysis of an annular diffuser with and without struts, Experimental Thermal and Fluid Science 22(2000) 183-195*

7. *Sebastian Muntean, Hakan Nilsson, Romeo F. Susan-Resiga, 3D Numerical analysis of the unsteady turbulent swirling flow in a conical diffuser using FLUENT and OPENFOAM, 3rd IAHR International Meeting of the orkgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, October 14-16, 2009, Brno, Czech Republic*

8. *Birajdar, H. S., & Pawar, M. S. (2013). Development of Rules for Method Selection for the Surfaces of Machining Cylindrical Part to Facilitate Computer Aided Process Planning (CAPP) for Jobbing Type Manufacturing Industries. International Journal of Research in Engineering & Technology, 1 (4), 13, 28.*

9. *Berge Djebedjian and Jean-Pierre Renaudeaux, Numerical and Experimental Investigation of the flow in annular diffuser, Proceeding of FEDSM‘98 1998 ASME Fluids Engineering Division Summer Meeting June 21-25, 1998,*

10. *Dr.N.Mohmed Sheriff, P.Selva Kumar, A.H. Syed Sulthan Alaudeen, CFD Analysis of flow in Afterburner, Proceedings of the 6th WSEAS International conference on Heat and Mass Transfer(HMT09).*

11. *Yogesh TV, Ganesan S, Kishore Kumar S, Effect of Exhaust Diffuser on Gas Turbine Afterburner Performance, Proceedings of the 37th National.*